

Injection moulding - The Mould

by R. Tomlinson – [Plastic Moulding Solutions](#)

Part design

You cannot start to talk about mould design without first talking about part design so here are a few useful tips when designing a product or part for the injection moulding process.

1. Keep it simple and avoid complicating things.
2. Have a list of the things that you want the part to be able to do or meet in other words a specification I call it my “must-have” list.
3. Consider what environment the part will be subjected to, what things may affect the part and the material like chemicals, temperature changes, fire, stress and most important standards or approvals that the material must adhere to like suitable for food use.
4. Quite often people design parts and then look for a material that fits within the must have list, use your must have list to eliminate materials and find the best fit material for the part at the start.
5. Polymer materials are always a compromise; what you gain in one area example strength you lose in another flexibility.
6. Polymer materials all have material specification sheets that inform you about the materials physical properties including what its application is designed for example caps, closures or thin wall containers.
7. Recycling of the part should always be considered during the design of the part when its end of life is reached.

Mould design

The basics:

1. Avoid large changes in wall thicknesses try to keep all wall thicknesses to a uniform thickness and follow the design principals regarding ribs, bosses and changes in wall thickness where possible. The thicker the part the more material is required to produce it and also the more heat requires extracting from the material, which means a longer cycle time both of which affect the final unit cost of the part.
2. Gate design and position (injection point) select the most appropriate point, which does not affect the filling of the part or leave an unwanted blemish or cosmetic scar on the part. Carry out mould flow analysis to arrive at the best solution and position.

3. Draft angles are required to help the moulding release from the mould be it the cavity or the core this is particularly important when designing the mould.
4. Radius should be used wherever possible to allow the material to flow into the mould polymers do not like very sharp corners or to be diverted in different directions.
5. Venting of the mould is essential to avoid moulding issues the material can only flow into the mould as fast as the air is expelled from the mould.
6. Cooling is a very important part of the mould it needs to be as close to the surface of the core and cavity as possible in order to remove the maximum amount of heat from the material.

Mould construction

Construction of the mould will depend a number of factors. Prototype moulds can be made from various materials including epoxy resin, printed 3D ABS, aluminium and steel it is usually determined by the requirements for a prototype part and the number of samples required for testing.

Construction of production moulds are generally determined by the following factors the annual requirements from the mould will determine the cavitation of the mould, this will determine the physical size of the mould, the complexity of the part, the cycle time requirements of the moulder, the lifetime of the product or part, the final cost of the mould and any guarantee given with the mould.

© Copyright R Tomlinson - Plastic Moulding Solutions 2019